American Scientific Research Journal for Engineering, Technology, and Sciences (ASRJETS)

ISSN (Print) 2313-4410, ISSN (Online) 2313-4402

http://asrjetsjournal.org/

Floating Wind Turbine Modelling for Analysis Using LS-DYNA3D Code

Mohamed Abdalla Almheriegh*

Assistant Professor, Department of civil Engineering, Faculty of Engineering - Tripoli University, P O Box 82677, Tripoli, Libya

Email: malmherigh@gmail.com

Abstract

Analysis is the crucial base to design, in complicated structure, this is more than vital, hence explicit finite element code LS-DYNA3D believed to be the perfect tool for such a hard task. Formulation of material, elements, and all environmental loads applied to various parts of the structure, as well as time step, are the corner stone for such analysis, the paper presents a suggested formulation for all aspects needed to run the proposed code. A floating full-scale wind energy converter chosen for analysis contributing to the simplification of a complicated structure in harsh environment on one hand and showing the superiority of the presented commercial code on the other.

Keywords: Analysis of wind converter; design of wind energy turbine; modeling of wind energy turbine; Floating wind converter; LS-DYNA3D code modeling.

1. Introduction

The aim of the work is to produce, if possible, a detailed mathematical representation of a floating offshore wind turbine using a suitable commercial 'state-of-the-art' finite element analysis code. Simpler models used in the past to estimate the global behaviour of such structures, but with time, the sophistication of analysis tools available to the engineer has increased. Hence, the paper will explore the possibilities for modelling the detail of such a structure, examine whether the detail is relevant, and give insight into the structural performance. The nature of the structure -shown after this section-, materials, boundary conditions and loading (both static and dynamic) will involve complexity, and the philosophy of how all of these aspects addressed is of the utmost importance.

---

* Corresponding author.

Explicit time integration techniques have proven to be an excellent tool for transient events such as impact, and some are highly developed in terms of modelling features. LS- DYNA3D is one of such codes and therefore been employed in this work. The key feature that the code should be able to model, is the floating behaviour of bodies. Main features that need addressed highlighted in the following sections:

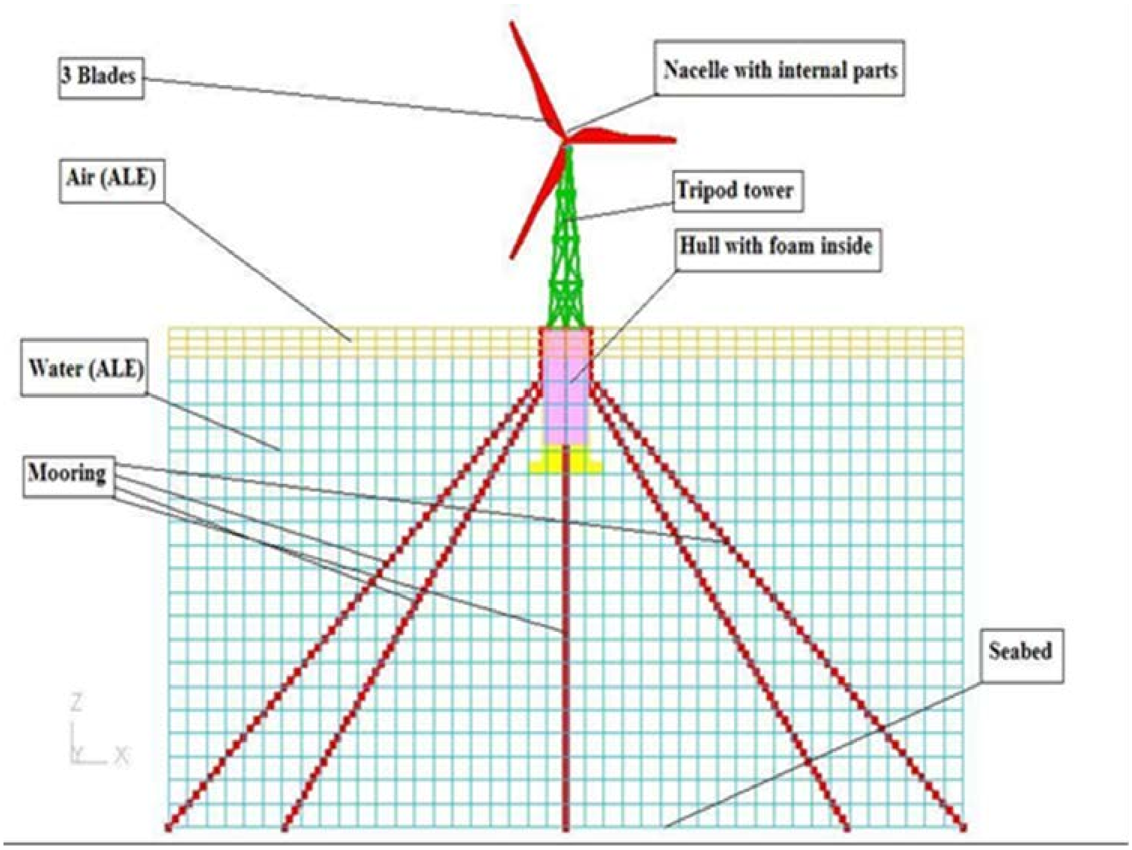

Figure 1

Floating wind energy converter

2. Model Features

2.1 Buoyancy Effects and Anchorage

The code must be able to model a buoyant structure floating in the sea. These potentially achieved in many ways.

a) Ignore any physical presence of the water and imitate the buoyancy effects by applying appropriate forces on the structure. The advantages of this are that they are easy to apply and are useful for the most simple of problems. However, the dynamic nature of the loading will be harder to represent in this way, as some prior knowledge of the float movement would be required. Water-structure interaction effects could possibly be modelled using springs and damper systems perhaps, similar to techniques used for soil structure interaction modelling for buildings.

However, the dynamic nature of the moving elements at the interface of very stiff (Lagrangian) concrete and the smooth (zero shear resistance) water elements is difficult to simulate. Furthermore, updating the stiffness of these springs accordingly and representing the sliding nature of the contact at the interface both are a difficult task. Numerical error provoked by adding a very stiff spring, either within the FE model or as part of the boundary 'Ill-conditioning' situation where the solution is very dominant by such very stiff element(s), Det Norske [1], Cook [2]. The 'pseudo' effects of the water forces on the structure modelled, however some of the important features of the water material such as dissipation of the pressure forces would be difficult to simulate.

b) The 'ALE' (Arbitrary Lagrangian Eulerian) feature available in LS-DYNA3D been used for many fluid-structure interaction problems, mainly for transient impact analyses where severe mesh deformation of the Lagrangian material presents a problem. To date though there appears to be little information specifically on modelling buoyancy effects and subsequent dynamics. Although we are not expecting the mesh to deform in this case, it could provide the appropriate buoyancy and sought fluid material effects. This could however be at the expense of increased elements and solution runtime, so model complexity will monitored carefully.

Typically, if using 'ALE', the turbine will have to float in a fluid 'half-space' of a size, which will best represent the effects of the pseudo-infinite sea. The boundaries will need careful consideration, and perhaps employ non-reflecting (energy absorbing) representation. It is this representation, which employed in this work, and a study of buoyancy conducted for verification purposes.

c) Floating turbines typically anchored to the seabed using thick polypropylene mooring cables. This anchorage best modelled using tension only spar elements, which are pinned to the model. In reality, the cables attached to the concrete turbine base through slip rings attached some distance from the top of the hull in a moment free arrangement near the top of the hull. This effect should be easy to model if required using the various nodal constraints available.

2.2 The Turbine Model

a) Given the mass and rigidity of the foam filled concrete hull, either modelled as a rigid material or assigned concrete material properties as appropriate. The foam core just serves to lower the mass centre of float modelled with the appropriate material and density.

b) The steel truss resides above the hull is assumed to comprise of rolled hollow sections, and may be modelled using simple beam elements.

c) The nacelle with associated internal parts will present a complex set of items to model depending on how relevant their real behaviour is to the response of the structure. Bearings and revolute gear all be modelled using the appropriate constraint conditions.

d) The blades will require modelling and can either be represented using beam or shell elements. These will serve to capture the aerodynamic loading is imparted to the rest of the structure. They made to rotate along the local rotor axis if appropriate.

2.3 Loading

There are many load effects to consider, and the way in which they are applied may have significant influence on the model behaviour.

a) Gravity will act upon the structure and will govern where the turbine floats in relation to the water, counteracted by hydrostatic forces. For simple force equilibrium representation, the gravity forces simply counteracted by an equilibrating pressure applied to the base of the structure. For the ALE representation, an equation of state needs to be specified which initiates the pressure within the fluid. This demands that a position of equilibrium reached after some time, once these initial loads applied (ramped on over a short time interval), i.e. bobbing of the structure witnessed.

b) Hydrodynamic forces from wave action may be present. These applied as static loads/pressures to the side of the turbine hull or possibly time varying loads if required. The use of the coupling between ALE grid and the Lagrangian mesh tends to account for creation of these contact pressures.

c) Aerodynamic loads applied as nodal force to the blades. A variety of wind speed scenarios exist which cause a non-uniform pressure distribution along the length of the blade governed by the 'Blade Element Method' and 'Momentum Method' theories equation. This accurately represented or perhaps applied uniformly if the non-uniform distribution found not to be significant through validation studies. Again, time varying load curves employed to investigate the dynamics and subsequent loading through the structure.

Finally, it is worth mentioning, that typically each load action normally applied at different times on the structure to confirm that its action properly represented. For example, wind loading applied some significant time after gravity and water pressure have had time to stabilise.

In summary, it is evident that there are a significant number of parameters considered in order to achieve a comprehensive model of the floating turbine. As all of these parameters addressed, this invariably results in an increase in model size with regard to the numbers of elements used and computer run-time. All of these parameters may affect the results obtained to a degree, hence, verification and validation work is essential to confirm the behaviour of each aspect of the model against well-known theory. Therefore, simpler models will enable us to have confidence that our representations are valid and taken forward into a more complex and comprehensive model if possible.

The sections that follow present aspects related to the validation work and procedures.

3. Lagrangian, Eulerian and ALE formulations

In general, the Lagrangian and Eulerian descriptions simply refer to the algorithm(s) of motion of material with respect to mesh of a quantum mechanics, while ALE (Arbitrary Lagrangian Eulerian) developed to combine both descriptions arbitrarily in a fashion reducing their drawbacks and enhancing their advantages.

- **Lagrangian algorithms**

In this case, each individual node of the computational mesh follows the associated material particle during motion and mainly used in structural and solid mechanics. It is allowing an easy tracking of free surfaces and interfaces between different materials, and facilitates the treatment of materials with history dependant constitutive relations. Its weakness is its inability to follow large distortions of the computational domain without recourse to frequent re-meshing.

- **Eulerian Algorithms**

Widely used for fluid dynamics modelling, the computational mesh is fixed and the quantum moves with respect to grid, in these descriptions, large distortions in the quantum motion handled with relative ease but at the expense of precise interface definition and the resolution of flow details.

- **(ALE) Description**

In this description, the nodes of the computational mesh may be moved with the quantum in a normal Lagrangian fashion, or hold fixed in Eulerian manner, or is moved in some arbitrarily specified way to a continuous rezoning capability. In ALE, greater distortions due to the free domain in movement could be handled than would be allowed by a purely Lagrangian method with more resolution than would be offered by a purely Eulerian approach. The ALE description is able to accommodate significant distortions of the computational mesh while preserving the clear delineation of interface typical of purely Lagrangian approach and conserving mass, energy and momentum. The ALE formulation works by re-meshing the material throughout the simulation so that the mesh stays relatively uniform, hence it will prevent local instabilities caused by highly distorted elements. Thus, ALE as proposed to bypass the difficulties arising from large deformation computations, which lead to numerical problems due to distortion of elements in classical Lagrangian formulation where mesh movement is attached to material movement. Buoyancy studies on simple model will illustrate and confirm how and whether the ALE is appropriate for use in this model.

4. Boundaries and Loads

4.1 Non-reflecting boundaries

LS-DYNA3D uses this option in solid brick elements only to model the indefinite domain (limit the size of the model). Boundaries defined as a collection of segments comprising the element faces. In half space, this option prevents artificial stress wave reflections generated at the model boundaries from re-entering the model and contaminating the results. Internally the code computes an impedance matching function for all non-reflecting boundary segments based on an assumption of linear material behaviour. Thus, the finite element mesh constructed so that all significant non-linear behaviour is contained within the discrete analysis model. In this algorithm, the infinite domain such as soil or seawater truncated via an artificial boundary, thus spurious wave reflection effects minimized.

Outside bounding nodes of the water 'ALE' part locked against translation to prohibit material from escaping thus reducing the pressure within it. While the non-reflecting boundary action achieved by creating, a segment set containing of the outer faces of all the bounding solid elements and the function 'non-reflecting boundary' evoked via this segment set.

4.2 Loads

Practically, the designer would consider that environmental loading experienced by typical structures comes from extreme values of all individual environmental parameters (wave, wind, current, etc.) that usually encountered. In reality, the likelihood that the maximum values of all environmental loads happening simultaneously is very remote. Therefore, since maximum values of environmental parameters do not occur at the same time, the application of joint probability of occurrence incorporated in determining the maximum loading exerted on the structure.

When considering the combination of wind and wave loads, the short-term wind climate is usually represented by the 10-minute mean wind speed 'U_{10}', and the short-term wave climate is usually represented by the significant wave height 'Hs', the two values usually interpreted as intensities of the corresponding wind speed and sea elevation process, respectively. The wind and waves, at a particular location usually have a common cause such as low pressure. Waves driven by wind and generated locally, at the same time; roughness implied by wave-affected sea surface influences the wind. Therefore, the two are the cause and effect of each other. Yet it is not reasonable to expect that the maximum wind speed will happen coincidently with the maximum wave height.

For design, it is reasonable to consider some relatively rare combination of wave and wind climate as the characteristic climate and then to find the maximum load response that occurs for this climate over its duration. In practice, usually the wave climate for the significant wave height with a 50-year reoccurrence period in combination with a wind climate conditioned on this wave e.g. the expected value of U_{10} conditioned on the 50-year significant wave height or some higher value of U_{10}. This methodology as suggested by Det Norske [1] and the UK Department of Energy [3] through [10]. The 100-year maximum occurrence period used instead, the two approaches are widely recommended throughout the literature Det Norske [1] and British Standards [11] through [15], in general the 50-year return period tends to be British practice, while the 100-year return period used in American and Norwegian practices.

For linear load combinations, 'Torstar's' rule plays a central role, Det Norske [1]. The rule states that the maximum value of the sum of two independent random processes occur when one of them has its maximum value. Application of this rule to the combination of two load processes, e.g. wave load and wind load implies that the combined load will be at its maximum either when the wave load is at its maximum or when the wind load is at its maximum.

Explicit finite element codes use time integration processes, therefore loads defined with time as load curves. For the purpose of verification, gravity loads should be applied confirming buoyancy behaviour. The buoyancy is very much dependant on geometry and mass loads as discussed in, Mohamed, [16]. Due to small inertia involved, and to reduce run time in the 2-D model -used and published for verification-, the gravity load applied ramped over a short time and the impact effect assumed be absorbed by the ALE damping action. Because of the size and mass of the 2-D model rendering it close to the neutral stable buoyancy case, no side loads therefore applied to it. In the 3-D small models as the inertia increases, the gravity applied ramped over a relatively longer period. In the 2-D model, referred to, a small characteristic nodal horizontal load of 1 KN (arbitrary chosen) is applied ramped on a set of nodes representing the supporting hull side opposing wave and current direction to simulate hydrodynamic action.

Surrounding hydrostatic water pressure on the submerged part is readily evoked via constraining the outer nodes and the application of the ‘equation of state’ and coupling contact namely ‘constrained Lagrange in solid’ function.

Wind loads on blades in the characteristic model are simulated by constant ‘10 N’ “arbitrarily chosen” nodal loads applied on the two blades at three nodes set intervals in the horizontal direction, simulating wind action. Detailed theoretical description for creating actual loads on a full-scale floating wind turbine was mentioned in, Mohamed, [16]. Meanwhile quantification considerations of these actual environmental service loads experienced by the full scale floating wind turbine in the form of load curves is also detailed in past reference.

5. Materials Models

LS-DYNA3D is equipped with a vast material library which is updated regularly by adding new material models, some of these models are still under verification and their use is largely left to the user, while some are defined for certain types of elements. While for the purpose of verification rigid material can perform well, material models suggested according to the expected action of the parts to which they assigned. Some parts ought to be rigid material such as the yaw mechanism, nacelle with internal parts, transmission and rigid wall for seabed modelling. These parts are straightforward to model and therefore, not listed herein while their material properties listed in the relevant sections. Rigid body material model defined for all elements used for structural purposes, keyword cards defining rigid material as given in, Mohamed, [16]. Description of the used materials models and reasoning for their choice given in the following bullets:

- **Modelling the water quanta**

For the floating wind energy converter hosting seawater is an integral part of the analysis and it is a prime driver for loading and structural behaviour. To simulate the behaviour the ALE brick elements are assumed acting in the finite quanta around and supporting the hull and the cable parts up to the specified height. This formulation assumes separate movement between Eulerian mesh ‘brick elements’ and multi-materials in dealing with fluid dynamics.

Material ‘type 9’ (*MAT_NULL) is typically defined by the code for fluids and is used which require the definition of an ‘equation of state’ to activate pressure inside fluid solid ALE elements. In addition, hourglass control card for fluid parts as defined, therefore the used cards are:

*MAT_NULL

*SECTION_SOLID_ALE

*EOS_GRÜNEISEN

The *Grüneisen* equation is a formula adopted by the code to initiate the pressure inside a compressed material. A MathCAD sheet adopting the use of this equation for water parameters as presented in, Mohamed, [16].

ALE 'multi-material formulation 11' used for which the seawater part needs to be defined as a 'multi-material group'; hence, a minimum of two 'ALE' materials should exist. The finite quanta modelled by applying a 'non-reflecting' boundary along the water cube sides. To initiate buoyancy pressure, build up due to gradual gravity loading, the outer nodes bounding the water body prohibited from translational movements to prevent the ALE material from "running away" thus reducing pressure within it. This done by defining a node set that contains all nodes in the bounding walls of the 'ALE' body.

To apply the coupling, Lagrangian parts are slaved to 'ALE' parts, which are the master in this contact type. Therefore, the float is in contact with the fluid 'multi-material group' (water and air parts). To achieve this coupling a keyword 'constrained Lagrange in solid' card defined. To guarantee full contact between the water 'ALE' part and the top air 'ALE' part, common nodes at their interface merged dictating that their common boundary needs to be coincident nodes.

An extremely useful option for 'multi-material group' is the capability of defining the initial volume occupied and thus updating the 'ALE' geometry keeping clear separation (material resolution) by updating the surface following the deformed shape of the multi-material mesh. Through re-meshing of the ALE deformed elements, this function used to accommodate the hull part inside the fluid parts in a way, which guarantees coupling action and saves tedious time consuming geometry creation, which could be very difficult in some configurations.

Therefore, to accommodate the hull part inside the fluid parts the used function is:

*INITIAL_VOLUME_FRACTION_GEOMETRY, it is used in creating three empty geometrical shape(s) representing the supporting hull. The value of 1025 kg/m^3 used for seawater density. The contact between the float part and the fluid 'ALE' group is done via the coupling algorithm of 'constrained Lagrange in solid', expanded in, Mohamed, [16]. Default element 'type 1' (one point integration solid element) used suggesting the necessary hourglass control need employed.

- **Modelling air quanta**

The air part is an 'ALE multi-material' part and intended for modelling air bounding the top surface of the water body in the formulation. Similar to the water part this part is modelled using material 'type 9' and 'ALE' solid brick elements 'formulation 11'. This dictates that air is defined as 'ALE multi-material group', therefore, fulfilling the condition of two materials need to be available for material group definition prerequisite for many functions to be used. An equation of state is also necessary for this type of material and therefore defined it is the 'linear polynomial', a density value of 1.1845 kg/m^3 assigned for air elements.

The 'linear polynomial' equation of state is used by the LS-DYNA3D code to initiate and govern the pressure in 'ALE' elements, typically for ideal gas.

In using this equation, the parameter (E0) defined as initial internal energy per unit reference specific volume needs to be calculated based on a given or assumed pressure in 'ALE' air elements. In this case, the 'E0' value is calculated assuming atmospheric pressure of (101, and 325 Pa) in air elements using the formula:

$P = C_0+C_1\mu+C_2\mu^2+C_3\mu^3+(C_4+C_5\mu+C_6\mu^2)$ which is the equation of state for linear polynomial, LSTC [17].

With $C_0=C_1=C_2=C_3=0$, $C_4=C_5=0.4$ and $\mu=1.8444e\text{-}05$, thus the parameter E0 is calculated:

101325 = (0.4+0.4x 1.8444e-05) x E0

Hence E0 = 253,307.8 Pa

The values of μ being air dynamic viscosity and other used coefficients detailed in, Mohamed, [16]. Therefore, required keyword cards for modelling air quanta are:

*MAT_NULL or 'type 9'

*SECTION_SOILID_ALE

*EOS_LINEAR_POLYNOMIAL for equation of state,

The constraints applied to this part are coupling with the water part, which done through merging the common coincident nodes at the interface and coupling with the slaved float part at their common interfaces. The coupling, which accounts for contact between the float and 'ALE' group done by keyword:

*CONSTRAINED_LAGRANGE_IN_SOLID. This unique algorithm for defining coupling (contact) in fluid-structure interaction is combined with the capability of calculating forces, and stresses in the 3 global axes of the structure as well as the average pressure on element or set of elements(s) segment(s). These forces, stresses and average pressure are activated only if the coupling between Lagrangian mesh and 'ALE' grid is defined, they are exported to the ASCII file 'd3plot' to be accessed and read by the post-processor and values are plotted. Further expansion of this useful algorithm discussed in, Mohamed, [16].

- **Modelling the float**

Typically, in the verification or debugging stage all materials are modelled as rigid body except the material to be verified. Rigid material 'type 20' provides a convenient way of turning one or more parts of any elements class into a rigid body. The LS-DYNA3D manual states that approximating deformable body as rigid is a preferred modelling technique in many real-world applications.

Rigid bodies are robust and versatile and very well developed in the algorithms involving their use in LS-DYNA3D such as contacts, constraints, imposed motions and displacements. The common nodes of both rigid body parts used in defining 'joint revolute' are coincident but not merged to capture the 'joint revolute' behaviour. Elements defined as rigid are bypassed in the element processing and no strains or stresses are calculated for them, consequently they are calculation cost (memory and time) efficient, hence they are preferred.

In rigid body material, all elements of the same material number are treated as rigid body, these elements are integrated to determine their common mass and centre of gravity and moment of inertia of the group, this group is dealt with as a rigid body with 6 degrees of freedom (3 rotations and 3 translations) applied to their common centre of gravity. The position of all rigid bodies is updated in LS-DYNA3D by a time integrator which works together with the central difference method of time integration (explicit method).

The float was modelled as a rigid body to confirm its buoyancy behaviour and then switched to elastic 'material type 1' with concrete properties since the real stresses are expected to be within the elastic range for the real supporting hull simulated by the float in this verification. The reason for switching to the elastic material model (model with finite stiffness) is the need to calculate the strains, stresses and forces for the elements involved. These are necessary physical values which are element based and not processed for rigid body elements. Employed material properties for concrete are expanded in, Mohamed [16].

- **Modelling the truss**

The material 'type 98' '*Material_simplified_Johnson_Cook' which is a simplified version of the material 'type 15' is used. This material model is defined for (beam) truss elements, and is primarily intended for structural analysis where thermal effects are insignificant. The hollow heavy steel sections and their connections intended for offshore structures are typically designed as plastic sections. This material model is cheaper to run, (although alternatively elastic material 'type 1' could be used instead). The material is not expected to exhibit plasticity, although it is included in the event that this could possibly occur, and 'beam section' for element property is used. Factors defined for steel in this material type as well as properties are mentioned promptly in the characteristic model to follow in this paper, and expanded in, Mohamed [16].

Only three material models are available for the truss (beam) elements, i.e. elastic 'material type 1', elastic plastic kinematic 'material type 3', or material 'type 098' Simplified Johnson Cook, in addition to the rigid body material. The 'Simplified Johnson Cook' material was primarily conceived for use in structural analysis where large strain rates may be experienced without the more advanced features of thermal or damage effects (50% faster solution than the full Johnson Cook). It could be argued that the elastic model or simpler plastic kinematic model would be suitable for use, and this point is accepted. But without fore knowledge of the behaviour of the system the employment of the simplified Johnson Cook plasticity model was judged to be appropriate in the event that large strain plasticity might occur, and an acceptable 'trade-off' in terms of its effects on solution runtime.

- **Modelling the cables**

The only material model defined by the code for cables modelling is material 'type 71', (*Mat_Cable _Discrete_Beam) specifically defined to model cable action, the material assumes elastic behaviour for section properties. Traditionally steel rope mooring will be the choice but due to the advantages of polyester taut ropes they will be chosen herein. These advantages include:

a) Polyester ropes weigh one-seventh less in-water than steel wire and reduce the weight of the mooring system by as much as 80 per cent. This leads to less vertical loads on the hull hence less size and less cost.
b) The use of polyester in the 'taught-leg' configuration is more effective in storing and releasing energy associated with fall and rise of wave, and the elastic response of the rope provides the restoring force to the hull. The straighter line to seabed reduces the length of the mooring by as much as 70 per cent directly lowering the cost of the mooring system and reducing seabed 'footprint'.
c) In taut-leg configuration, the angle at which the mooring line is connected to the hull is much larger (as measured from vertical). A larger component of the line tension therefore acts horizontally counteracting the wind, wave and current forces.
d) Polyester ropes are superior to steel wire in fatigue characteristics which is one of the critical aspects in vibrating structures.

The primary function of the mooring cables is to keep the body in position under the drifting action of wind, current and waves.

The eight-mooring line choice is based on offshore engineering past experience, so that the buoy is kept in place even if one of the lines is lost. The mooring choice is adaptable to water depths between 75m and 500m. The catenaries are of taut wire synthetic fibre rope for the whole water depth with the section passing through guiding shoes made from steel cables. Due to the cost and difficulty in erecting and anchoring piles, more than one mooring cable share the same anchor pile in the multiunit wind farm. Keyword cards associated with this material include element property and beam section as follows:

*MAT_CABLE_DISCRETE_BEAM and

*SECTION_BEAM

The top and lower ends of these cables are restrained against translational movement but not rotation this is readily defined via the keyword 'constrained interpolation'. More than one neighbouring node from the hull and seabed parts to the cable termination node(s) is used in this constraint to avoid local stress concentrations. Cables are coupled to surrounding 'ALE' water elements through the algorithm 'constrained Lagrange in solid' as already discussed in previous bullets. Cross section properties used in the verification are mentioned promptly later in the relevant part while further expansion is given in, Mohamed, [16].

- **Modelling the blades**

The blades are represented by material ‘type 3’ material plastic kinematics and of glass reinforced plastic ‘GRP’ properties. GRP dominates the market because it provides the necessary properties at low cost. The important characteristics of GRP are good mechanical properties, good corrosion resistance, high temperature tolerance, ease of manufacture, and favourable cost. Typical values are (170Mpa, 100Mpa, 170Mpa and 12E+03Mpa) for tensile strength, compressive strength, bending strength and modulus of elasticity, respectively Det Norske Veritas, [1]. Blades are modelled with plate shell fully integrated elements formulation ‘16’ to avoid ‘hourglass’ energy in the system. Material ‘type 3’ is selected due to the overstressed condition of the blades once in service at the extreme wind conditions especially at the blades roots where the fluctuating (negative and positive) stresses are expected to reach plastic capacity in this region. Rigid material could be used but will offset the possibility of calculating the element stresses. Properties of GRP used is expanded in, Mohamed, [16].

6. Running the Solver

- **Negative volume termination errors**

There is no obvious answer to the chronic problem of error termination due to the difficulties in identifying certain reason behind it. The problem is more complicated when more than one reason is involved. However, there are certain precautions might help in debugging, if not preventing these errors. There is in general a deficiency of error reporting in most of the finite element codes. In LS-DYNA3D some errors are reported especially those related to rigid bodies or constraints and material requirements while some are not. The most common solutions for negative volume errors can be addressed as follows:

a- Use the most updated release of the code (LS-DYNA3D).
b- Stiffen up the stress-strain curve for the material involved.
c- Double check units and material properties for relevant parts.
d- Turn all damping off.
e- In foam 57 increase damping parameters to the max. 0.5
f- Use *control_interior for soft materials (foam)
g- Turn logic off in *control_contact and ERODE=1 in *control_timestep
h- Reduce time step scale factor in *control_timestep
i- Avoid fully-integrated solids (formulation 2 or 3), which tends to be less stable in large deformations, instead use default (type 1) element with type 5 or 6 hourglass control, or tetrahedral solid elements with element formulation 10 for soft foams.
j- Use *contact_automatic and reduce bucket sort factor.

- **Using dynamic relaxation**

It was discussed earlier that LS-DYNA3D has limited capabilities in performing static analysis. For dynamic problems, the dynamic relaxation feature is used to initialise equilibrium (initial) state. During such transient analysis both mass and damping matrices lose their physical background and become fictitious quantities to control the iteration process. Physically this method amounts to the imaginary situation of immersing the structure in a viscous fluid which damps the strong geometric nonlinearities. To start with a few runs were carried out on both the verification models and the full-scale model to assist the use of this algorithm, however, it’s use was excluded to reduce run time and because the kinetic energy difference (the criteria for dynamic relaxation convergence tolerance) was found to be small enough from the first few cycles. This suggests that convergence was either attained in early stage. Dynamic relaxation will continue to run unless a time limit to terminate it is imposed and the analysis continues until the convergence is successful. When dynamic relaxation was excluded the models were seen to equilibrate satisfactorily using the normal dynamic formulations, and a decision was made to use loads applied in the normal transient phase only.

- **Hourglass control**

Hourglass energy is noisy and erroneous and results in mathematical conditions that have no physical meaning and therefore need to be controlled. LS-DYNA3D assumes a default value type 4 unless modified via the hourglass control card. Due to high deformations inherited with the ALE, hourglass control is necessary for default ‘type 1’ one point integration elements, hence this is defined as type 4 with an hourglass factor of HQ = 0.0005 to guard against instability (defined and called in the part definition). Further discussions aim at prevention of these modes, and more expansion is given in, Mohamed, [16].

7. Damping

Energy dissipation is damping, it causes the amplitude of free vibration to decay with time, and limiting the amplitude of vibration produced by a loading whose frequency coincides with a natural frequency (resonance). Damping can be inherited (material) or algorithm added (artificial), in structural dynamics response damping can be:

- Viscous damping exerts forces proportional to velocity which appears as the damping matrix in the dynamics equation. Viscous damping is supplied by surrounding fluids or be added to the structure.

- Inherited damping is a material effect resulting from small plastic effects while the material is still within the elastic range. Other forms are friction or contact damping typically termed as Coulomb damping.

- Radiation damping refers to energy loses due to practically unbounded quanta such as soil and seawater like (infinite medium).

In LS-DYNA3D damping is introduced as a default value that can be changed where it is needed. In some material models such as foam and in all contacts damping is mandatory. Fluid/structure interaction coupling (contact) also has the effect of dissipating energy, as it is adopts the penalty type of contact which uses artificial spring stiffness for penalty force calculation. The radiation damping effect is also present via the non-reflecting boundary algorithm intended to simulate the infinite where energy is dissipated or absorbed thus prohibited from re-entering the system. The algorithm (artificial) damping choice is also available in the code via the readily defined dynamic relaxation function. In LS-DYNA3D, system damping can be applied at any time during the solution either globally via damping keyword card(s), control card, or on the defined material keyword card. The calculation of factors involved in the damping calculation, by no means is an easy task, for over damping will result in prolonged analysis with all burdens of computer time and memory, while under-damping results in numerical noise (erroneous). In this verification the control damping card is activated to allow default damping values.

- **Run time and time step**

The time step is crucial for the stability, convergence and accuracy of explicit analysis. LS-DYNA3D automatically chose the critical time step for the system based on the maximum frequency of the smallest element and uses this step throughout the analysis. This is what is known as conditional stability, i.e. time integration is stable if the time step decreases.

The time step is limited by stability and it usually falls during the analysis due to the elements being distorted, (it could also rise as well). It is not possible to use a time step larger than the critical time step but it is possible to use a lower time step which will increase the run time (but has been shown not to improve the results).

The time step is reduced by reducing the default 'SCFT' value (0.9) on '*control_timestep' keyword card, or by specifying a time step versus time load curve on the same card. Instability shown by rapid rising energy and a float over flow error will occur if the period of any mode of deformation in the model is less than π times the time step.

Roughly speaking, the time step Δt = 0.9 L/C for any element, L being the smallest distance between any two nodes and C being the sound speed in the material (the beam cross-section and shell thickness are not considered when calculating time step). If the time step does not conform to this criterion, computed displacements and velocities grow without limit, thus causing instability and error termination. In most of the problems the time step selected by the code is good enough for accurate analysis. The code uses different formulas for different types of elements employed based on element geometry properties, Young's modulus, Poisson's ratio and material density, then the smallest time step is used for analysis.

For lengthy simulations where the number of time steps goes above half a million or so, double precision executable version of the code must be used to minimize the numerical round off errors.

The time step calculated by the code were used in the verification models while using other possibilities of reducing the time step is advisable for the full model where the saving in time can be useful due to the number of DOF involved. The LS-DYNA3D code adapts two possibilities for improving the run time by adjusting the critical time step calculated by the code using one of the following two approaches:

i. Mass scaling technique

In this approach, the same time step (larger than the code calculated) is used for all elements by adjusting the density of elements. This is activated by positive 'DT2MS' on '*control_timestep', or adjustment applied to elements with time step less than a specified value and activated by giving 'DT2MS' on the same keyword card negative value. Mass-scaling carries a burden of having to confirm that the addition of non-physical mass does not significantly affect the results (the same is said for time scaling).

ii. Sub-cycling

In this technique elements are grouped and stored based on their time step (mixed time integration). This will speed up the analysis run time and allow local mesh refinement which is efficient when the element size in the mesh varies significantly. This approach is a mixed time step process where the time step is used for each group thus avoiding the use of the smallest one and improves the run time. Further important issues in running the solver are:

- One point integration element reduces the amount of effort required to compute the strain matrix by 25 times over the 8 point integration element, while in the element strains and the nodal forces calculations the number of multipliers is reduced to 16 times for the one point integration element over the 8-point integration one.
- In the solution of the plasticity problems and problems where Poisson's ratio is close to 0.5, fully integrated elements can experience constant volume bending modes.
- Time integration is intended for finding numerical solutions for the nonlinear differential equations representing the equations of equilibrium for the nonlinear finite element system. This numerical solution is cheaper and easier to attain than an analytical solution.
- The time step need to be small enough to assure that the stress wave does not propagate across more than one element per time step.
- When imposing loads, load curves are gradually ramped up from zero to avoid the excitation of high frequency response caused by abrupt impact effects. This practice is observed throughout the process of creating load curves.

- **Mesh size**

Finite element analysis is the simulation of a physical approximation of the real system. Using simple, interrelated building blocks called elements; a real system with infinite number of unknowns is approximated with a finite number of unknowns, since the FEA model is the mathematical idealization of the real system.

In the real world, no analysis is typical, as there are usually facets that cause it to differ from others.

The art of using the FEM lies in choosing the correct mesh density required to solve a problem. If the mesh is too coarse, then the element will not allow a correct solution to be obtained. Alternatively, if the mesh is too fine, the cost of the analysis in computing time can be out of proportion to the results obtained. In order to define a relevant mesh, some idea of the parameter distributions (stress, pressure, etc.) within the component is required. If the answer is known, then a good mesh can be defined. A fine mesh is required where there are high parameter gradients and strain whereas a coarse mesh is sufficient in areas that have stress resultant contours of reasonably constant slope.

A good or appropriate mesh is one that enables accurate resolution of the underlying physical phenomena, yet is coarse enough to allow a fast solution time.

Element aspect ratio is the ratio between the longest and the shortest element dimension. Acceptable ranges for aspect ratio are element and problem dependant, but users are normally given limits such as 3:1 for stresses and 1:10 for displacements. There is no hard and fast rule, the limit to aspect ratio can be affected by the order of the element, displacement function, numerical integration used and the material behaviour.

The two mesh factors that affect accuracy are element quality and mesh density. Not all the features in a model will need the same level of accuracy and so a variation in mesh density through the model is usually appropriate; striking a balance between required accuracy and effort required in adjusting the mesh and time to run it.

LS-DYNA3D has the option of re-meshing and adaptivity that could be invoked via the control cards or some material cards. The manual however, elaborates that these options are still under development and testing, therefore their use was excluded. Verification model(s) preferably ‘2-D’ must be done before the full scale model is run. The above guidelines of aspect ratio, the ease of defining coupling (contact), as well as time, pressure and stress criterion must be observed in mesh construction. Also, one to one correspondence of the nodal elements at the contact interface has to be achieved, thus guaranteeing full contact. Pressure and stress values ought to show good agreement giving confidence in the mesh size and the value of 0.01 default (FEMB 28) element size can be used in the 2-D model. This meshing if arrived at after a few trials is therefore judged to be sufficient. In the full model and due to the size and number of elements involved fine meshes has to be observed at critical locations only while smooth transfer in mesh size is satisfied.

8. Conclusions and Future Work

- The LS-DYNA3D code is for sure capable of modelling the floating offshore wind turbine in terms of geometry, constrains, materials, different loads applied and boundary conditions that employed to capture complications involved in contacts. Minimizing the lateral movement of the floating structure will increase energy capture and reduce fatigue effects but expected to increase local stress. Improving the aerodynamics of both the supporting structure and the superstructure is vital in reducing the effects of the applied loads and hence cheaper design.
- Because the center of mass of a horizontal axis wind turbine is quite high, a massive structure is necessary to support the wind turbine. Hydrodynamic stability is achieved by moving the center of mass as low as possible through placing the ballast in the lowest possible location this will lead to further elongation of the support structure which minimize the heave motion due to wave action.

- The location of some parts: E.g., transformer part (if needed) does not need to be in the nacelle. Therefore, locating such parts somewhere; either, at the seabed or at the floating surface of the turbine will contribute good deal to reducing the weight of the superstructure, thus lowering the center of gravity of the whole assembly and consequently increasing the restoring moment and improving the buoyancy stability.

- Firm debugging regime must applied through 2-D modelling as well as 3-D chacteristic smaller model, while the first done and published showing excellent match, the second currently under consideration and running it will be good future work. Of concern also as further work, running the full model as well as investigating fatigue effect of the dynamic loads applied to the structure. Building the model in another parallel explicit code and run it, could serve as future work as well and alleviates level of confidence.

References

[1] Det Norske Veritas “Guidelines for Design of Wind Turbines” second edition 2002, printed by Judsk Centraltrykkeri, Denmark ISBN 87-550-2870-5

[2] R D Coock et al “Concepts and Applications of Finite Element Analysis” John Wiley & Sons 2002.

[3] Department of Energy “Probability Distributions for Environmental Loads on Offshore Structures” report prepared by Lloyd’s Register of Shipping for the Department of Energy, offshore technology report OTI 88 504 HMSO publication 1988 UK.

[4] Department of Energy “INFOIL 2 THESAURUS TERMINOLOGY FOR THE OFFSHORE INDUSTRY” offshore technology information OTI 88 505 HMSO publication 1988 UK.

[5] Department of Energy “Probability Distributions for Environmental Loads on Offshore Structures” report prepared by Lloyd’s Register of Shipping for the Department of Energy, offshore technology report OTI 88 504 HMSO publication 1988 UK.

[6] Department of Energy “INFOIL 2 THESAURUS TERMINOLOGY FOR THE OFFSHORE INDUSTRY” offshore technology information OTI 88 505 HMSO publication 1988 UK.

[7] Department of Energy “Metocean Parameters–Parameters Other Than Waves” offshore technology report OTH 89 299 HMSO publication 1990 UK.

[8] Department of Energy “Guidance on the Design and Construction of Offshore Installations” HMSO publication 1974 UK.

[9] Department of Energy “Metocean Parameters–Wave Parameters” Offshore Technology Report OTH 89 300 HMSO publication 1990 UK.

[10] Department of Energy “Fluid Loading on Fixed Offshore Structures Volume I” Offshore Technology Report OTH 90 322 HMSO Publication 1990 UK.

[11] British Standard “Wind turbine generator systems-Part 13: Measurement of Mechanical loads, DD IEC TS 61400-13, first edition, 2001-06, British Standards online. http://www.tionestop.com/tionestop/where/bs.htm, January 2003.

[12] British Standard “Wind turbine generator systems-Part 23: Full-scale structural testing of rotor blades, technical specification, DD IEC TS 61400-23:2002 first edition,2001-04 British Standards online.http://www.tionestop.com/tionestop/where/bs.htm, June 2003.

[13] British Standard “Wind turbine generator systems Part 2: safety of small wind turbines” BS EN 61400-2:1996 IEC 1400-2:1996, IEC online British Standards website.

http://www.tionestop.com/tionestop/where/bs.htm, July 2003.

[14] British Standard "Wind turbine generator systems-Part 11: Acoustic noise measurement techniques" BS EN 61400-11:1999, IEC 61400-11:1998, November 1998, British Standards online. http://www.tionestop.com/tionestop/where/bs.htm, June 2002.

[15] British Standard "Wind turbine generator systems- Part 12: Wind turbine power performance testing" BS EN 61400-12:1998, IEC 61400-12:1998, April 1998, British Standards online. http://www.tionestop.com/tionestop/where/bs.htm, August 2002.

[16] Mohamed A Almherigh, "Evaluation of Finite Element Techniques Applied to Floating Offshore Wind Turbine" Ph. D thesis, The University of Salford, Salford, UK, 2006.

[17] LSTC "LS-DYNA Keywords User's Manual" April 2003, Version 970 Livermore Technology Corporation, PDF online, LSTC website. http://www.lstc.com/ , July 2003.